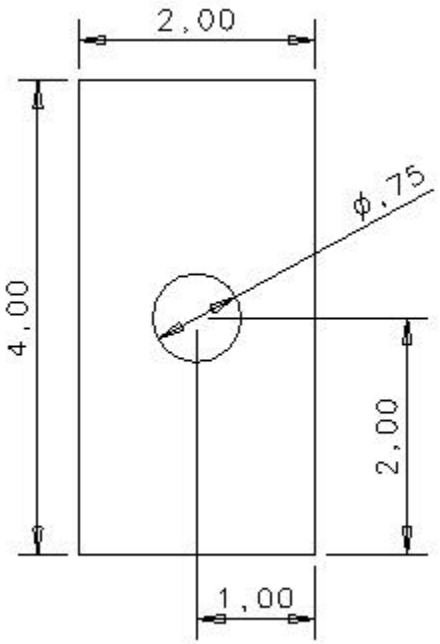
Creating a Sheet Metal Part

Panel – Panel Method

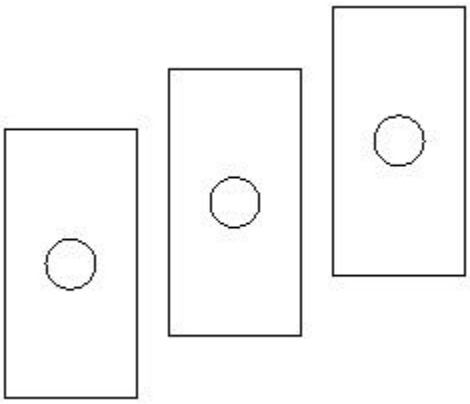
This method allows panels to be created separately then combine the panels to create a sheet metal part. Sheet Metal will add the bends to the panels and all the bend-panel operations will be available in the history.

1. Create the geometry shown below. Specific dimensions and constraints are not important for this demonstration.



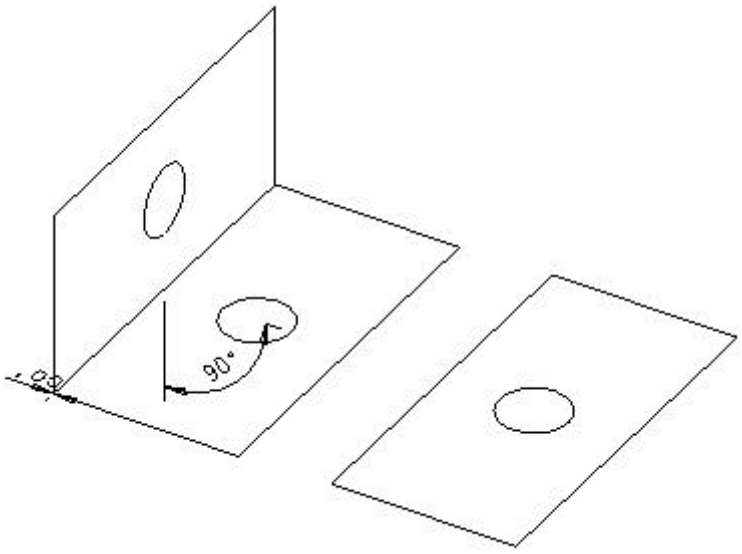
2. Open the Sheet Metal subpanel found under row five, center column, at the bottom. Select Build Panel and select all of the wireframe. This creates a flat surface, or panel.

3. Make two copies of this panel.

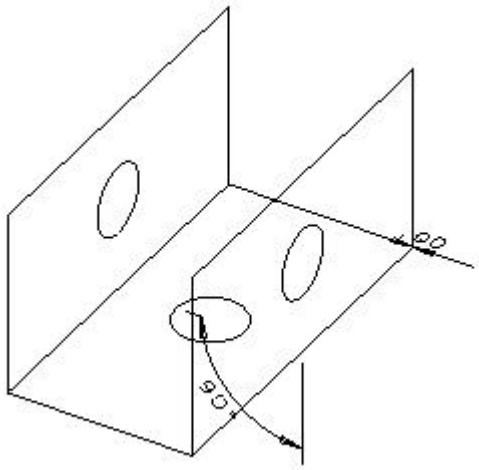


4. Select Create Bend from the subpanel.

5. Pick one of the long edges on one of the panels. The first panel picked is the one that is moved into position. Pick the long edge of another panel. This one remains stationary. Once the panel is in position you can enter a different angle, negate the angle to be on the other side of the stationary panel, flip the panel end to end, or slide the panel along what will become the bend. Situate the first panel as shown below.

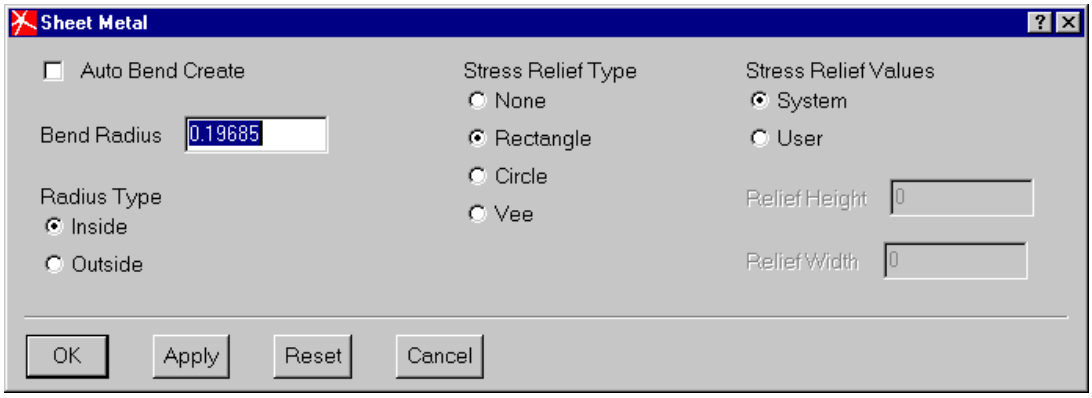


6. Situate the second panel as shown below.

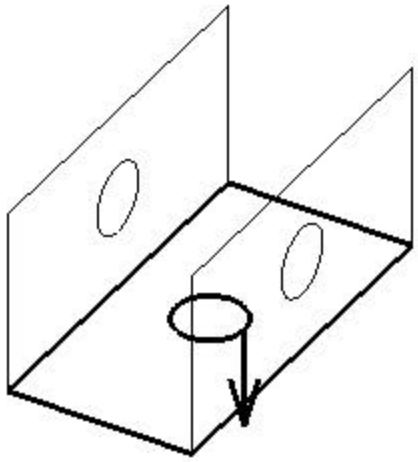


7. Select Sheetmetal from the subpanel and pick the part.

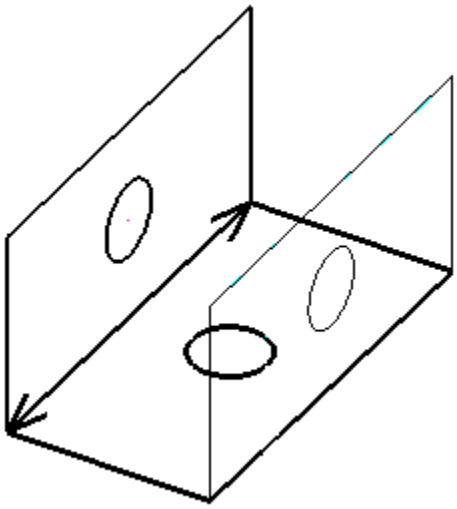
8. Accept the default material. The Sheet Metal form automatically appears. Notice that you have several options specific to sheet metal creation. Accept the defaults for this demonstration.



9. Pick the surface shown for the Ground Panel. This is the panel that remains stationary when the part is unfolded. It also is the panel that determines thickening direction for the part. Pick Yes to accept the direction shown, No to change the direction.

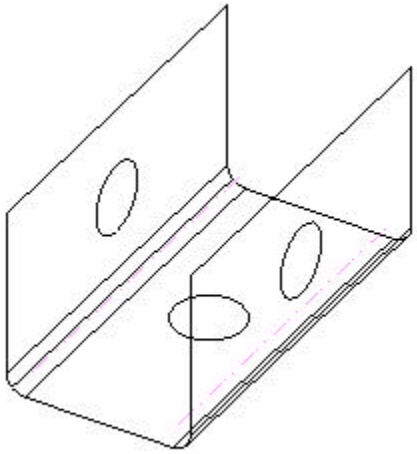


10. Accept the default the software begins asking if bend locations are correct.

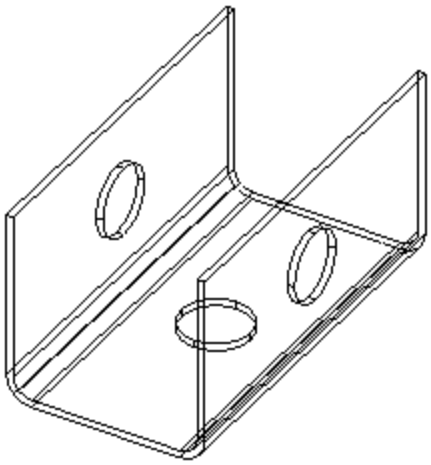


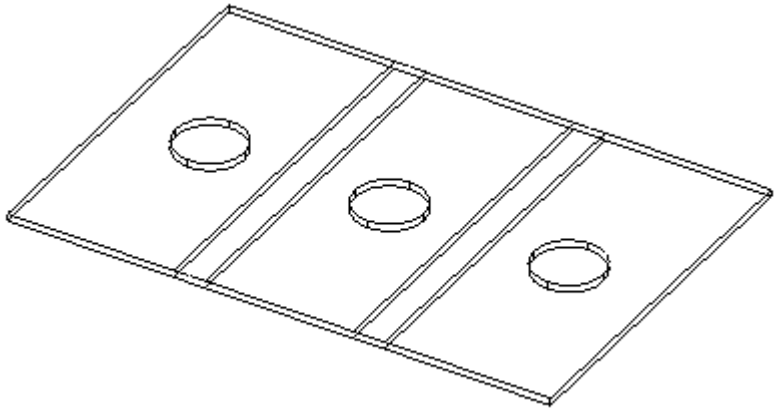
At each bend you have individual sheet metal options in case a particular bend has different characteristics than what was defined on the Sheet Metal form. The prompt asks you to select a modification from the pop up list. Accept the defaults for this demonstration. When all bends are defined the software will offer you the opportunity to add more bends. It will then offer the opportunity to add more panels.

The resulting part will appear as below.



11. Shell the part. Notice on the Shell form that the options for thickness and direction are unavailable. These are determined from the material selected and the thickening direction specified in step 10.



The part is now a sheet metal part and can be folded and unfolded.

12. Use Fold and Unfold Panels to show a preview of the new state. Pick Update to complete the task.